Journal of Physics: Conference Series

PAPER • OPEN ACCESS

Study of natural convection development in narrow vertical channels

To cite this article: E V Starikov *et al* 2018 *J. Phys.: Conf. Ser.* **1105** 012005

IOP Conf. Series: Journal of Physics: Conf. Series **1105** (2018) 012005 doi:10.1088/1742-6596/1105/1/012005

Study of natural convection development in narrow vertical channels

E V Starikov, V A Klimova and S E Shcheklein

Department of Nuclear Power Plants and Renewable Energy Sources, Ural Federal University named after the first President of Russia B.N.Yeltsin, 620002, 19 Mira street, Ekaterinburg, Russia

E-mail: s.e.shcheklein@urfu.ru

Abstract. The process of heat transfer due to natural convection in narrow vertical water-filled pipes is considered. Experimental and simulation data are given. The mechanism of natural convection development is analysed.

1. Introduction

It is well known that natural convection in enclosed spaces, for example, in narrow vertical tubes of various heat engineering equipment, influences its operating features. In particular, in thermosyphons (heat pipes) a slug regime of boiling appears in the case of natural convection lowering, which reduces the heat transfer and decreases the thermosyphon efficiency [1–2]. In consequence of this it is required to reveal the dependency of natural convection intensity on the inner diameter of the pipes and the way of heating.

The present research considers the heat transfer in a vertical circular channel filled up with water and composed of two parts. The lower part is heated and the upper part is cooled. Heat transfer is implemented in expense of thermal conductivity and natural convection.

According to [3], natural convection influences the heat exchange in vertical pipes in case of the Rayleigh number Ra = Gr•Pr > 10^3. The traditional approach to the impact of natural convection in the heat exchange means determination of the Rayleigh number and the average Nusselt number as

$$Ra = \beta g \rho^2 C_p L^3 \Delta T / (\lambda \mu), \; Nu_{av} = q_{av} L / (\Delta T \cdot \lambda) \quad (1)$$

where β is the volume expansion coefficient, g is the gravitational acceleration, ρ is the fluid density, C_p is the specific heat at constant pressure, L is the characteristic length-scale of the system, ΔT is the temperature difference between hot and cold 'walls', q_{av} is the heat flux from the wall to the fluid, averaged over the wall, λ is the thermal conductivity, μ is the dynamic viscosity of liquid.

The application of this approach in the analysis of natural convection in narrow vertical channels is limited by the following reasons:

- The characteristic dimension, which is usually determined in channels as a layer thickness between the hot and cold walls, in this case depends on the channel size. We can't unambiguously choose the pipe diameter or length or their ratio as a characteristic dimension.
- The choice of the determinant temperature and the temperature difference for the heat-transfer coefficient calculation also depends on the channel characteristics.

IOP Conf. Series: Journal of Physics: Conf. Series **1105** (2018) 012005 doi:10.1088/1742-6596/1105/1/012005

2. Experimental study

The complex research in laboratory environment was carried out to study the process of fuel boiling in enclosed space. The researchers used the glass pipes with a diameter from 4 to 30 mm that were filled up with distilled water. In course of experiments the values of supplied heat flux, the pressure change in the channel, and the fluid temperature were measured. The detailed description of the experimental facility and the research procedure is given in [4] and [5].

The characteristic feature of steam generation in these conditions is almost periodic emergence of steam pistons and their migration to the upper part of the channel above the liquid face with subsequent destruction as a result of condensation. Supercooled condensate trickles down to the zone of heat supply (evaporator) where it is heated again together with the remaining fuel until the next moment of heat generation. This results in the appearance of the self-oscillating process of pressure change in the channel, which intensity is defined by the properties of the boiling liquid, constructive and operation conditions of the process. A dependency of measured pressure pulse amplitude and frequency on the heat flow density was observed at the self-oscillating process in the narrow channels with a diameter of 7…24 mm. The influence of power of heat supplied to the evaporating part of the channel reveals in the form of formation of single steam pistons with q = 6 kW/m^2, developed self-oscillating process with q = 13 kW/m^2 and small-scale pressure pulse typical for bubble flow with q = 22 kW/m^2.

The conducted research does not give a possibility to study the heat transfer mechanism in a narrow vertical channel and to explain the impact of natural convection into the heat exchange process. That's why a computer simulation using the commercial software Solidworks Flow Simulation was carried out to study the case.

3. Creation of the computer model

3.1. The computer model and boundary conditions

A computer model is a pipe filled with water (figure 1). The lower part of 180 mm length has the constant heat supply in the form of specific heat flux. The upper part of the pipe of 360 mm length provides the heat elimination to the environment. Here a constant inner wall temperature was set equal to 20°C. Gravity was also considered during the simulation in the form of gravitational acceleration g directed along the channel axis.

The channel diameter varied from 4 to 30 mm. Specific heat flux was from $5{,}9\cdot10^3$ to $29{,}5\cdot10^3$ W/m^2. The dimensions of the model were chosen in accordance with the characteristics of experimental facility [4, 5].

Such features as heat conduction in solids and capability of phase transition were excluded in the simulation. Thus, water stayed liquid regardless of the temperature.

The pipe wall was not included into the computational domain because the aim of the research was to study the convection inside the vertical channel. Also it simplifies the problem because the wall thickness and material varies in existing installations.

3.2. The governing equations and numerical solution

The commercial software Solidworks Flow Simulation was used in this study. This software simulates hydrodynamics and heat exchange using the Raynolds-averaged Navier-Stokes equations in the form of mass, angular momentum and energy conservation laws in Cartesian coordinate system:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial t}(\rho u_i) = 0 \tag{2}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i u_j) + \frac{\partial p}{\partial x_i} = \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ij}^R) + S_i, \quad i = 1,2,3 \tag{3}$$

$$\frac{\partial \rho H}{\partial t}+\frac{\partial \rho u_i H}{\partial x_i}=\frac{\partial}{\partial x_i}(u_j(\tau_{ij}+\tau_{ij}^R)+q_i)+\frac{\partial p}{\partial t}-\tau_{ij}^R\frac{\partial u_i}{\partial x_j}+\rho\varepsilon+S_i u_i+Q_H, \tag{4}$$

where u is the fluid velocity, ρ is the fluid density, $S_i = -\rho g_i$ is a mass-distributed external force per unit mass due to a buoyancy, where g_i is the gravitational acceleration component along the i-th coordinate direction, $H = h + u^2/2$, h is the thermal enthalpy, Q_H is a heat source or sink per unit volume, τ_{ik} is the viscous shear stress tensor, q_i is the diffusive heat flux. The subscripts indicate summation over the three coordinate directions.

These equations are supplemented by fluid state equations, that define the nature of the fluid, and by empirical dependencies of fluid density, viscosity, and thermal conductivity on temperature. The same system of equations is used for both laminar and turbulent flows and for transient flows.

The finite volume method is used to discrete the governing equations. Rectangular axis-oriented cells are built in the fluid volume. Near the solid/fluid boundary, the mesh is obtained by cutting original parallelepiped cells that intersect the geometry. The near-boundary cells are polyhedrons with both axis-oriented and arbitrary faces as a result of this approach.

All the calculated physical parameters are defined in the mesh cells' centers. The obtained cells are refined during the calculation process in accordance with the gradient of physical parameters. The mesh refinement is carried out in several stages before and during the calculation.

In this study the most accurate of the possible mesh levels was used. Furthermore, the extra accuracy of curvilinear faces refinement was set. Figure 2 shows the initial mesh in the horizontal section of the model.

To validate the ability of Flow Simulation to predict the features of such flows the designers offer to examine natural convection in a square cavity filled with air with given temperature difference between the vertical walls. The left and right vertical walls were held at the constant temperatures of 305 K and 295 K, accordingly, whereas the upper and bottom walls are adiabatic. The length of the cavity between cold and hot walls was taken as a characteristic dimension. It varied within the range of 0.0111...0.111 m in order to vary the cavity's Rayleigh number within the range of $10^3\ldots10^6$. The results of simulation in the form of the average Nusselt number dependency on the Rayleigh number showed good coincidence with the empirical data of [6].

4. Simulation results

4.1. Presence of natural convection

Numerical simulation showed that there is a significant heat drop between the heat supply and heat elimination zones in the channels of a low diameter. Figure 3 shows the values of dimensionless

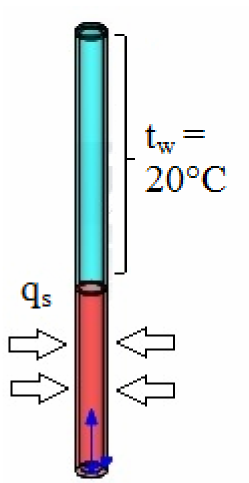

Figure 1. Computer model.

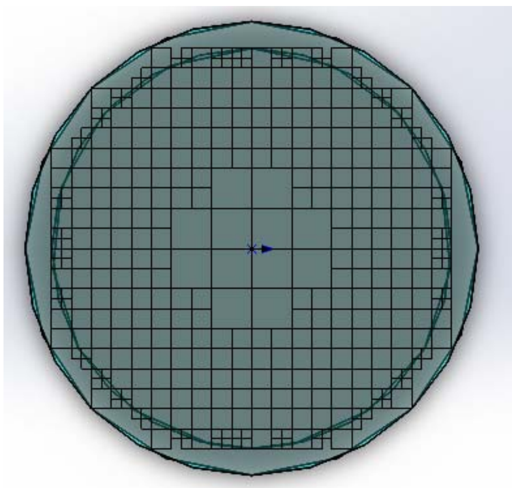

Figure 2. Initial calculation mesh.

IOP Conf. Series: Journal of Physics: Conf. Series **1105** (2018) 012005 doi:10.1088/1742-6596/1105/1/012005

temperature at the channel axis for various diameters with supply of 5.9 kW/m^2 specific heat flux. The dimensionless temperature was calculates as the ratio of current temperature value to the temperature in the lowest point of the channel. One can see in the figure 3 that the temperature in the heated and the cooled zones is almost uniform for the channel of 30 mm diameter due to the intensive heat mixing in vertical direction. The channels of a low (less than 12 mm) diameter are characterized by the following: heat elimination occurs at the relatively low area; the liquid is heated up to high temperature in the heated zone. It indicates that the heat transfer in the upward direction is complicated.

The results obtained correlate with the experimental data of [4]. In the course of the experiment the slug regime of boiling was not observed in the channels with diameter of 16 mm and larger. As it was said, the presence of slug boiling indicates the difficult heat transfer in vertical direction.

The results achieved allow to conclude that the critical diameter in this case is 12–16 mm. It should be considered that the results were obtained for water and for heated area height of 180 mm. In general, the presence or absence of heat transfer in vertical direction depends on the ratio of heated area length and channel diameter.

4.2. Study of heat transfer along the channel height.

Now let's consider in detail the mechanism of heat transfer in the channel.

The calculation showed that vertical flows appear in the channels of a relatively large diameter (more than 20 mm). In the area of heat supply the liquid moves in upward direction near the wall and descends in the near-axis part of the channel. In the area of heat removal we can observe the opposite situation: descending current near the wall and up-going movement in the center of the channel (figure 4). Temperature distribution in the axial and near-wall areas shown in table 1 gives the possibility to take the channel axis as a second "wall" being cold in the heat supply area and hot in the

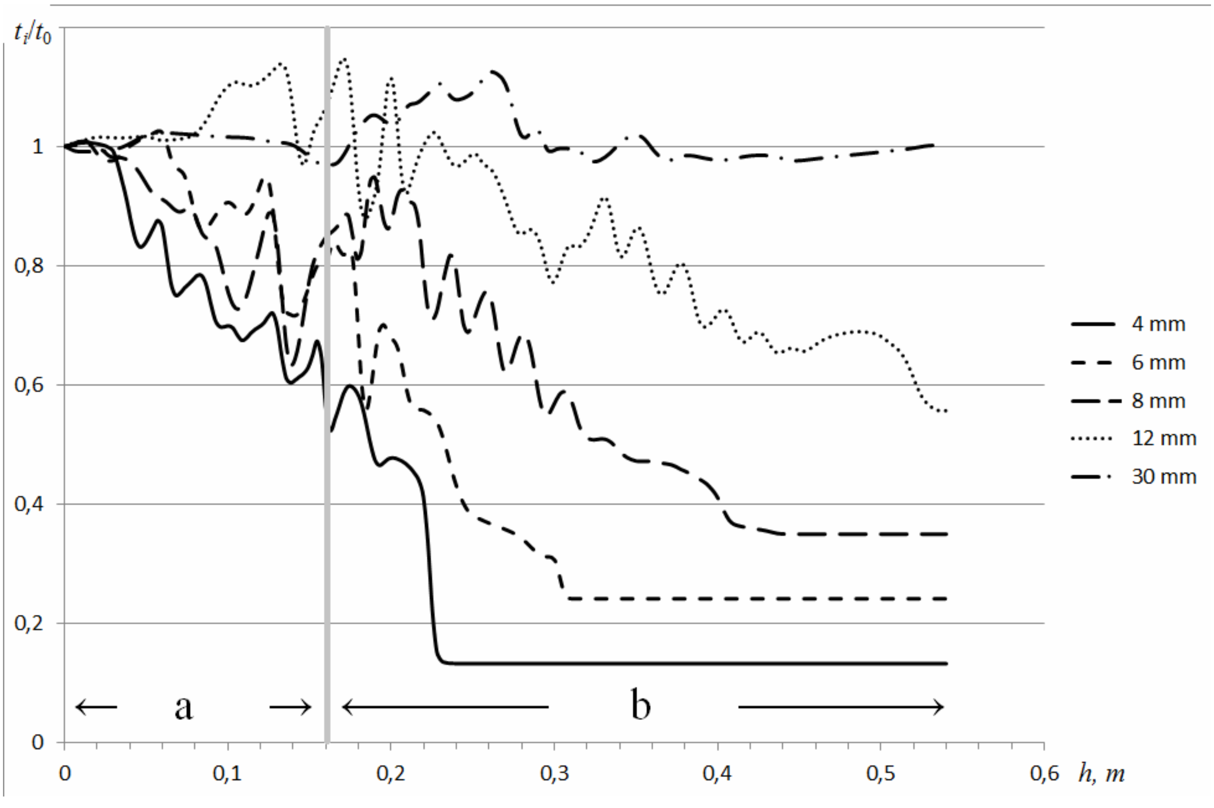

Figure 3. The values of dimensionless temperature t_i/t_0 along the channel height h for different diameters with q = 5,9 kW/m^2: a – «hot» area; b – «cold» area.

IOP Conf. Series: Journal of Physics: Conf. Series **1105** (2018) 012005 doi:10.1088/1742-6596/1105/1/012005

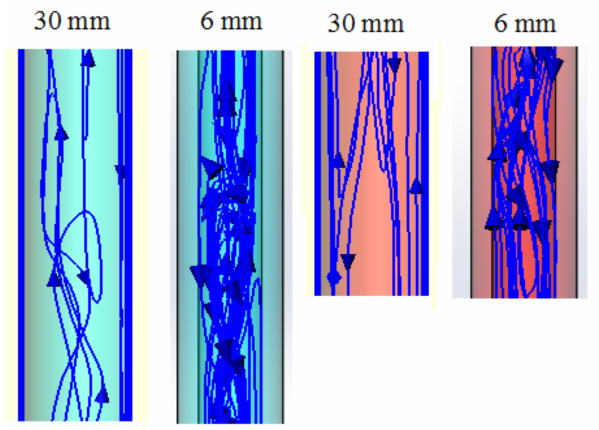

Figure 4. Flow trajectories in the heat removal area (left) and heat supply area (right) in the channels with diameter of 30 and 6 mm. Proportion is not followed.

heat removal area. One can use for such channels the traditional calculation methods that were mentioned in the Introduction, dividing the channel into several parts in vertical direction and using the channel radius as the characteristic length-scale of the convection.

The channels of lower diameter show the other picture. The flow trajectories in both hot and cold areas form the structure which resembles a twisted cord. The transfer of liquid to the upper part of heat removal area is practically cut off. For example, the flow trajectories in the channels with the diameter of 30 and 6 mm are shown in figure 4. The trajectories were built from several points in the boundary between heat supply and removal zones.

Figure 5 shows the velocity component directed along the channel axis for the channels of a relatively low and big diameter. These data confirm the conclusions made with the flow trajectories. In the relatively large channel we can observe the currents in opposite directions in near-wall and axial areas. In relatively narrow channels the upward and downward velocity directions alternate in both areas.

Table 1. Average temperatures in different parts of the channels, °C.

	Heated area		Cooled area	
	channel axis	near wall	channel axis	near wall
d = 4 mm, q = 6 kW/m^2	112.6	113.3	20.46	20.10
d = 4 mm, q = 11,8 kW/m^2	195.9	190.3	21.04	20.30
d = 6 mm, q = 6 kW/m^2	75.3	82.0	23.18	20.69
d = 8 mm, q = 6 kW/m^2	48.1	60.0	27.43	21.54
d = 10 mm, q = 6 kW/m^2	44.1	56.1	28.47	21.41
d = 12 mm, q = 6 kW/m^2	38.54	44.58	28.19	20.61
d = 12 mm, q = 11.8 kW/m^2	63.07	74.22	28.22	21.36
d = 12 mm, q = 17.7 kW/m^2	66.66	80.19	31.72	21.30
d = 18 mm, q = 6 kW/m^2	31.86	40.06	27.80	20.94
d = 18 mm, q = 11.9 kW/m^2	39.32	54.26	30.87	22.21
d = 24 mm, q = 6 kW/m^2	28.71	37.11	27.54	21.34
d = 24 mm, q = 11.9 kW/m^2	38.03	47.78	31.21	22.43
d = 24 mm, q = 17.7 kW/m^2	39.70	56.45	32.90	24.15
d = 30 mm, q = 6 kW/m^2	26.61	31.72	26.49	22.72
d = 30 mm, q = 11.9 kW/m^2	32.98	40.30	29.97	25.38
d = 30 mm, q = 17.7 kW/m^2	38.85	51.47	34.05	26.83

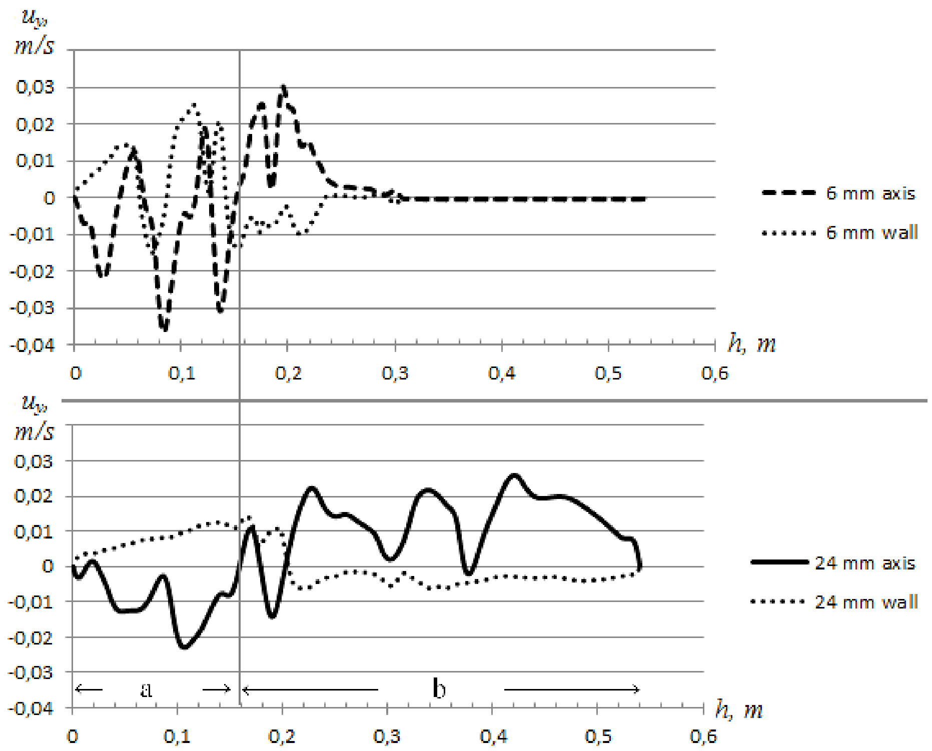

Figure 5. The velocity component in vertical direction in near-axis area for 6 and 24 mm channels, q = 5,9 kW/m^2.

5. Conclusion

The simulation of natural convection in narrow vertical pipes showed that the convection intensity depends on the inner diameter of the channel and the heat flux supplied. The conditions of natural convection development are determined by the constraints that prevent the generation of currents in vertical direction.

The results of simulation coincide with the experimental results of [4, 5]. In both studies natural convection is not developed in the channels with a diameter less than 16 mm in case of specific heat flux from 5.9 to 20 kW/m^2.

The results obtained can be used for optimization of heat transfer and improving of efficiency of closed single-phase thermosyphones.

References

[1] Bezrodnii M K, Pioro I L and Kostyuk T O 2005 *Processy Perenosa v Dvuhfaznyh Termosifonnih Sistemah Teoria i Practica* (Kiev: Fakt)

[2] Pokusaev B G, Pribaturin N A, Mesarkishvili Z S and Shchetinskii O Yu 1989 Heat exchange in condensation of a single steam piston *Izvestia Sibirskogo otdelenia AN SSSR Seria techn. nauk* **6** 3

[3] Kutateladze S S 1990 *Teploperedacha i Gidrodinamicheskoe Soprotivlenie: Spravochnoe Posobie* (Moscow: Energoatomizdat) p 173

[4] Starikov E V, Pahaluev V M and Shcheklein S E 2008 Possibility of Thermomechanical Conversion of Solar Energy *Alternativnaya Energetica i Ekologiia* **11** 97

[5] Burov A V, Starikov E V, Pahaluev V M and Shcheklein S E 2010 Use of Low-potential Heat Sources for Supply of Autonomous Energy Accumulators *Promyshlennaya Energetica* **6** 33

[6] Technical reference. Solidworks Flow Simulation 2015.